SPECIAL REPORT 91-8

CRREL

(2)

Simulation of Oil Slick Transport In Great Lakes Connecting Channels

User's Manual for the Microcomputer-Based Interactive Program

Poojitha D. Yapa, Ronald J. Thomas, Jr.,
Randall S. Rutherford and Hung Tao Shen

July 1991

For conversion of SI metric units to U.S./British customary units of measurement consult ASTM Standard E380, Metric Practice Guide, *published by the American Society for Testing and Materials, 1916 Race St., Philadelphia, Pa. 19103.*

This report is printed on paper that contains a minimum of 50% recycled material.

Special Report 91-8

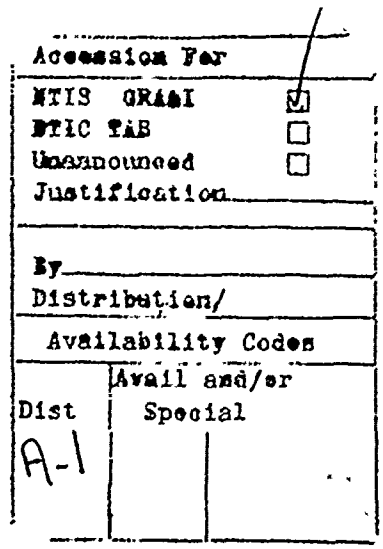
U.S. Army Corps of Engineers
Cold Regions Research & Engineering Laboratory

Simulation of Oil Slick Transport In Great Lakes Connecting Channels

User's Manual for the Microcomputer-Based Interactive Program

Poojitha D. Yapa, Ronald J. Thomas, Jr.,
Randall S. Rutherford and Hung Tao Shen

July 1991



Prepared for
OFFICE OF THE CHIEF OF ENGINEERS

PREFACE

This report was prepared by Poojitha D. Yapa, Assistant Professor of Civil and Environmental Engineering, Clarkson University; Ronald J. Thomas, Jr.; Randall S. Rutherford, Raytheon Co.; and Hung Tao Shen, Professor of Civil and Environmental Engineering, Clarkson University. The study was supported by the U.S. Army Corps of Engineers under Contract No. DACA33-85-C-0001. Steven F. Daly and Michael Ferrick of the U.S. Army Cold Regions Research and Engineering Laboratory are the contracting officer's technical representatives. The writers thank them, as well as Dan Thompson and Don Williams of the Detroit District, U.S. Army Corps of Engineers, for their cooperation and assistance throughout the study period. Jay B. Perry assisted in changing the program to run under VGA resolution and compiling it under Turbo BASIC.

The writers also acknowledge the assistance provided by the following individuals during various stages of this study: J.R. Weiser and R. Thomas, Detroit District, U.S. Army Corps of Engineers; F.H. Quinn and D.J. Schwab, Great Lakes Enviromental Research Laboratory, NOAA; J. Galt and T. Raiser, NOAA; M. Sydor, Inland Water Directorate, Canada; G. Tsang and R.O. Ramseier, Environment Canada; D. Mackay, University of Toronto; J.A. McCorquodale, University of Windsor; and S. Venkatesh, Atmospheric Environment Service, Canada.

This report is one of a series of reports on numerical simulation of oil slicks in inland waterways. The series coordinator is Steven F. Daly, CRREL.

CONTENTS

ILLUSTRATIONS

Figure

TABLES

Table

Simulation of Oil Slick Transport In Great Lakes Connecting Channels

User's Manual for the Microcomputer-Based Interactive Program

POOJITHA D. YAPA, RONALD J. THOMAS, JR., RANDALL S. RUTHERFORD
AND HUNG TAO SHEN

INTRODUCTION

The growing concern over the possible impacts of oil spills on aquatic environments has led to the development of a large number of computer models for simulating the transport and spreading of oil slicks in surface water bodies. Almost all of these models were developed for coastal environments. With the increase in inland navigation activities, oil slick simulation models for rivers and lakes are needed.

Two computer models, named ROSS and LROSS, have been developed for simulating oil slick transport in rivers and lakes, respectively. The study was originated by the Detroit District, U.S. Army Corps of Engineers in relation to the Great Lakes limited navigation season extension study. The oil slick transformation processes considered in these models include advection, spreading, evaporation and dissolution. These models can be used for slicks of any shape originated from instantaneous or continuous spills in rivers and lakes with or without ice covers. Although developed for the connecting channels in the upper Great Lakes, including the Detroit River, Lake St. Clair, the St. Clair River and the St. Marys River, these models are site independent and can be used for other rivers and lakes.

The programs are written in FORTRAN programming language to be compatible with the FORTRAN77 compiler. In addition, a user-friendly, menu-driven program with graphics capability was developed for the IBM-PC AT computer, so that these models can be easily used to assist the clean-up action in the connecting channels should an oil spill occur.

This report is one of four volumes, which together provide a complete description of the analytical formulation of the models, the logic and structures of the computer programs, and the instructions for using the models (Shen et al. 1990, in prep. a,b).

This manual is written to provide users with the necessary instructions to use the oil spill simulation models interactively on an IBM-PC AT computer. The interactive program, named MICROSS, can perform a variety of tasks, including creating data files and running the computer programs for ROSS or LROSS, plotting results on the screen and obtaining plots on the attached HP plotter. These plotters can be made available in both monochrome and color versions. In the color version the distribution of oil can be classified into eight colors. In the monochrome version the oil slick is represented by a distribution of dots. While every effort has been made to make this program as self-explanatory as possible on the screen, first-time users should read this manual before using the program. This manual will guide you step by step through the program, while providing some additional information that is not readily available on the screen. It is expected that after one or two trial runs, you will be able to use the program without consulting this manual. The hardware and software requirements for this program are given in Appendix A.

STARTING THE PROGRAM AND CHOOSING OPTIONS

The programs were originally developed when IBM-AT keyboards were different from the ones available today. Although MICROSS has been significantly revised and improved in 1990, some restrictions that existed in the original versions still exist. One of these restrictions is the need to have the "Num Lock" key on during graphical operations.

The programs are supplied on a floppy disk in archived form. To install the programs on your hard disk, copy all files to your subdirectory first. Then type "Install" and hit "Enter." In a few minutes MICROSS will be installed on your computer. Type "MICROSS" and hit "Enter" to start the interactive model. The screen will then display the following:

```
                    OILSPILL SIMULATION MODEL
                          (ROSS/LROSS)

        Developed by the Department of Civil and Environmental Engineering
                  Clarkson University, Potsdam, New York 13676
        under the support of the Detroit District, U.S. Army Corps of Engineers, through the Cold Regions
                  Research and Engineering Laboratory, Hanover, N.H.

This program is furnished by the Government and is accepted and used by the recipient upon the express
understanding that the United States Government makes no warranties, expressed or implied,
concerning the accuracy, completeness, reliability, usability, or suitability for any particular purpose of the
information and data contained in this program or furnished in connection therewith, and the United
States Government shall be under no liability whatsoever to any person by reason of any use made
thereof. The program herein belongs to the Government. Therefore, the recipient further agrees not to
assert any proprietary rights therein or represent this program as other than a Government program.

Hit any key to continue.
```

To continue with the program, hit any key. Then the following will appear on the screen:

```
                    This is the main menu for the Oilspill Simulation Programs.

NOTE: LROSS is for Lake St. Clair and Detroit River

          ROSS is for:  St. Clair River
                        Detroit River
                        Upper St. Mary's River
                        Lower St. Mary's River

1. Create data files IPROCD.BAS
2. LROSS with output on the screen
3. LROSS with output on the printer
4. ROSS with output on the screen
5. ROSS with output on the printer
6. Graphic Routines IPROGR.BAS
7. Exit

Hit the number of the desired option:
```

As indicated in this display, there are seven options to select from. These seven options correspond to different combinations of tasks as shown in the flow chart in Figure 1.

If option 1 is selected, you will enter the program to create data files for ROSS or LROSS. This program is named IPROCD (Interactive PROgram for Creation of Data files) for easier reference.

After all the data files are created, the computer will return to the MAIN MENU. At this point you may choose one of the seven options in the MAIN MENU for the desired operation. If option 3 is selected, you will execute LROSS or ROSS using data files in the computer, with output being directed to the printer. In this case you must ensure that the printer is properly connected and turned on. The power light, the ready light and the "On Line" light on the printer must be lit. If any of these three lights is not lit or the "Paper Out" light is lit, consult the printer manual. At the termination of execution the computer will return to MAIN MENU. Options 2 or 4 execute LROSS or ROSS using data files in the computer, with output on the screen. At the termination of the execution the computer will return to MAIN MENU.

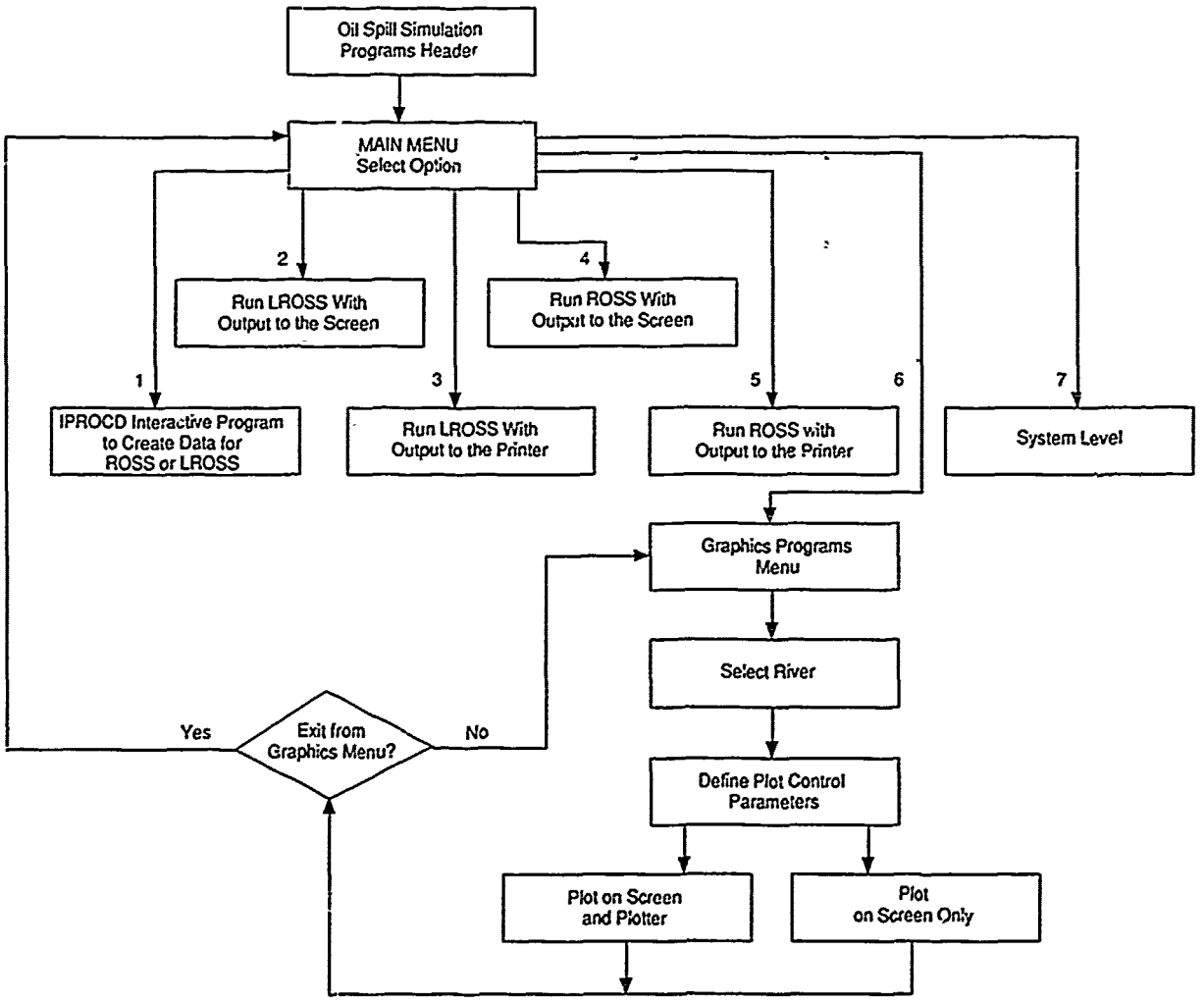


Figure 1. Flow chart for the menu-driven program.

Option 6 of the MAIN MENU is to execute the graphics and/or plotting routines. These routines were developed for displaying oil spill simulation results graphically. They can also perform a variety of other tasks. The graphics programs are described in more detail later in this manual.

The seventh option in the MAIN MENU is to exit from the MAIN MENU and return to the DOS (system) level. This option is for experienced users only.

CREATION OF DATA FILES

Determining file names

The first step in the data creation program IPROCD is to enter the names of the data files to be created. Upon entering IPROCD the screen will display:

```
This program interactively creates the input data files necessary to run the Fortran programs ROSS and
LROSS.
Enter the first 6 characters which defines the filenames. The unit numbers will be automatically assigned
by the program.

> STCL11
```

Names for data files that are needed for a ROSS or LROSS run all consist of a name formed by not more than six characters and a three-character extension, i.e. YYYYYY.xxx. The first portion is common for all data files in a specific computer run. The extensions are defined by the program to be compatible with the requirements of ROSS and LROSS. The user can define the first six characters. The output file from ROSS/LROSS where plotting data will be stored has a name starting with four or five characters defined in the table below plus SP.OUT. For example, if you are running the program for the Detroit River, the output file will be DETRSP.OUT. This will become the input file for plotting programs. In selecting the file name, it is recommended that the first four or five characters be used to identify the river or lake systems, as given in the following table:

First four or five characters: River/Lake

STCL: St. Clair River
DETR: Detroit River
STMU: Upper St. Marys River
STML: Lower St. Marys River
LDETR: Lake St. Clair and Detroit River.

The fifth and sixth characters (for lake runs, only the sixth character) can be used to identify the run number. For example, the eleventh run for the St. Clair River will have STCL11.xxx, where xxx is the extension that will be added by the program based on the type of data. These characters are used to save data files (if any) that will be created during data creation. After typing in the file name, hit "Return." If you entered STCL11 the screen will now display.

```
The six characters entered are: STCL11
Is this correct? (Y/N)
```

This gives you a chance to correct a mistyped name. If the six characters entered are correct, hit the "Y" key.

IPROCD can create data files for either ROSS or LROSS programs. There are slight differences between the data files of ROSS and LROSS.

After you answer the question in the above section, the following will appear on the screen:

> Will these data be created for ROSS or LROSS? (R/L):

The program asks for which program the data is to be created for, i.e. ROSS (River Oil Spill Simulation) or LROSS (Lake and River Oil Spill Simulation). You should select the oil spill simulation model appropriate for the spill site, and answer this question by hitting "R" for ROSS or "L" for LROSS.

Creating data files

The program IPROCD has several subsections. Each subsection will create one of the data files that will be used by ROSS or LROSS. The program allows you to create these data files one by one. You may skip any one of the data files if you already have one with the desired data. This feature is executed by answering "Y" or "N" in each subsection for the data file to be created. The details of these subsections and instructions for creating the data files are given in the following sections.

Subsection SPL

The first data file has the extension SPL. This file consists of two blocks of data. The first block contains oil characteristics and simulation parameters. The second block contains wind and temperature data. You may choose to create this file when the following appears on the screen:

> This segment of the program creates the file with the extension SPL. (This file contains oil characteristics, simulation parameters and weather data.)
>
> Would you like to proceed in this section? (If you enter No, this data file will not be created.) (Y/N):

If you wish to create or change this data file, hit "Y;" hit "N" if you wish to skip creating this file.

This is the first question in this section:

> What type of oil?
>
> 1. Gasoline
> 2. Bunker C
> 3. Fuel Oil No. 2
> 4. Other

To answer this question, just hit the desired option number. If you enter 1, 2 or 3, your answer will be used later in this subroutine for determining oil characteristics. The parametric values that define oil characteristics will be displayed at the appropriate occasion on the screen, but you can override and change them. If you select option 4, the program will prompt you to enter the parametric values that define characteristics at appropriate places.

For example:

> What is the type of oil?

Suppose you answered kerosene. The program will give you a chance to make a correction by displaying.

> The oil selected is: kerosene
> Is this correct? (Y/N):

If this is correct, type "Y;" otherwise type "N."
The rest of the discussion will assume that you selected oil type 1, which is gasoline. After typing in this response, hit "Return." The following will appear on the screen:

```
The oil selected is: Gasoline
Is this correct? (Y/N):
```

If a mistake was made, hit the "N" key. If the type of oil is correct, hit the "Y" key. Then a series of seven questions will follow:

```
What is the total time of oil spill simulation in hours?
    Note: This value must exceed the time step in unsteady flow model, i.e. in FLW file.
    If you plan to use any of the default FLW files the answer to this question must exceed 24 hrs. If this
    causes inconvenience you can create FLW data files with a smaller time step.
What is the frequency of obtaining output from PLOTNU and other subroutines, i.e., "1" for every step, "2"
for every other step. [1]?
Would you like to have cross section geometry data and shore conditions data (written to a separate file.)
(0-No/1-YES): [0]?
Would you like an output of particle locations to a datafile to be used in plotting (0-NO/1-YES). [1]?
Would you like a printer/screen number plot of oil slick particle distribution (refer to reference manual)
(0-No/1-YES): [0]?
What is the duration of oil spill (hrs.): [1.0]?
Do you wish to use the default formulation for the horizontal diffusion?
For river?
Hit the "Return" key for the default formulation or enter the desired value for horizontal diffusion
coefficient. (sq. ft/sec)
```

The first question asks for the total time of the oil spill simulation in hours. To answer this question, type in the time and then hit "Return."

The second question asks for the frequency of obtaining output from the subroutine PLOTNU and other subroutines. PLOTNU plots represent the oil concentration by the distribution of the number of particles in each grid. For example, inputting a value of 2 will give output every other time step. Other outputs include the spill information on the screen or printer and data file created for later use in plotting. The data file used for plotting tends to become large for longer simulations. The size of this data file can be reduced by not writing information every time step. The default for this question is 1, as shown between the brackets. If the default value is to be used, hit the "Return" key. Otherwise, type in the selected value and hit the "Return" key.

The third question asks if an output of fixed data (cross-sectional geometry and shore conditions) is desired. If this is answered with a 1, the output will be written to a separate output file. An answer of 0 means this output file will not be created. To answer this question, hit the "Return" key to accept the default value, or type in the selected value and hit the "Return" key.

The fourth question asks if an output of particle locations to a data file to be used in plotting is desired. The answering pattern is similar to that of the previous question.

The fifth question is to determine if a printer or screen number plot is desired for particle distribution. The output destination (printer or screen) depends on which choice is selected in the main menu. If this question is answered with 1, a number plot such as the example shown in Figure 2 will be generated during the execution of ROSS or LROSS.

The sixth question is to find the duration of the oil spill. This variable determines whether the spill is to be treated as continuous or instantaneous.

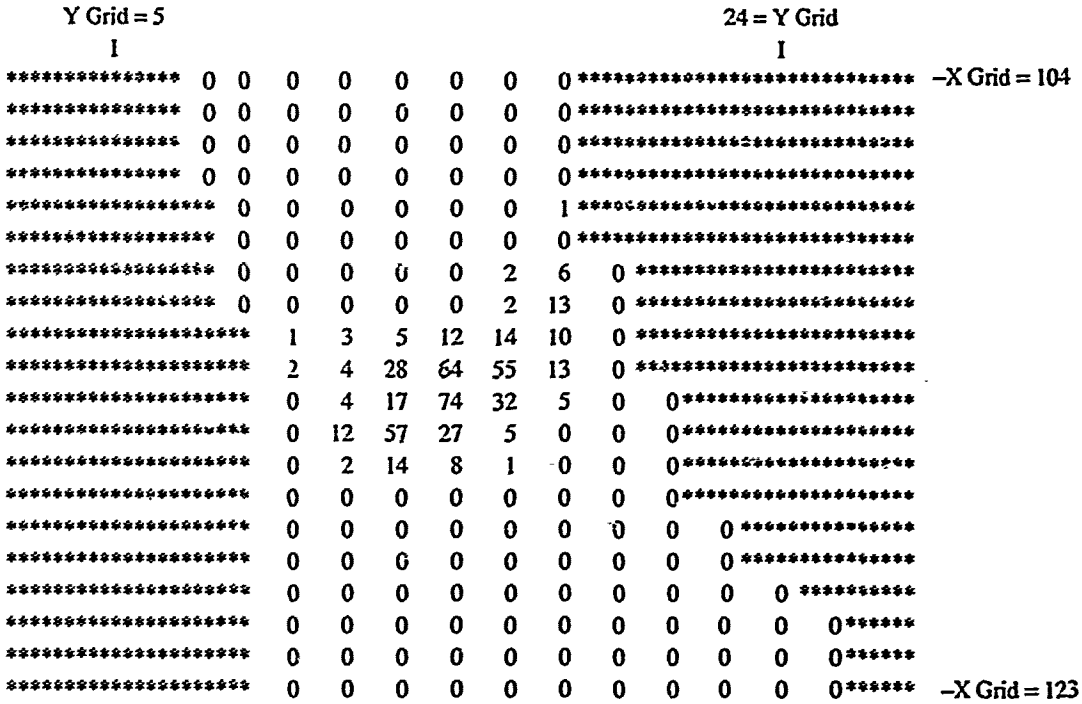


Figure 2. Sample output from the PLOTNU subroutine

ROSS and LROSS allows two options for computing the horizontal diffusion coefficient. The seventh question allows you to choose the option desired. To answer this question, just hit the "Return" key to accept the default formulation, or type in the horizontal diffusion coefficient and then hit the "Return" key. If you are creating data for LROSS, you will have to enter the diffusion coefficient for the lake in addition to one for the river. The questions and answers in both cases are similar. [The details of the default formulation are given in Shen et al. (1990).]

After answering this series of questions, you will be given a chance to determine if you have entered the correct numbers. The following will be displayed on the screen for verification:

Total time of oil spill simulation (hrs.): 4

Frequency of obtaining output form PLOTNU and other subroutines
(Number of steps between outputs): 1

Output of fixed data cross section geometry and shore conditions
(0-NO/1-YES): 0

Output of particle locations to a datafile to be used in plotting
(0-NO/1-YES): 0

Printer/screen number plot of particle distribution
(0-NO/1-YES): 0

Duration of oilspill (hrs.): 1.0

River-Default formulation for the horizontal diffusion coefficient has been selected.

Is the above information correct? (Y/N):

If the information that the program printed on the screen is correct, hit the "Y" key. If the information is incorrect, hit the "N" key. If you answered "N," the series of questions will be repeated and you will have a chance to correct the mistakes

Next a series of questions will be asked to determine spill characteristics. The list of questions is on the next page.

```
What is the total number of particles defined in the system? [500]
(Maximum allowed is 1000)

What is the total volume of the oil spill (U.S. gal.)?

What is the length of time step for simulation?: [15] min

What is the specific gravity of oil? Gasoline [0.7]

What is the kinematic viscosity of water? [ 141.100E-07] sq ft/sec

What is the surface tension of oil? Gasoline [0.7550E-03] lbs/ft
```

The first question is for specifying the number of particles to be used to represent the oil slick. The default value is 500, as shown inside the brackets. If this value is desired, just hit "Return." If a different value is desired, type in the value and then hit "Return." If you typed in a number larger than 1000, the maximum value of 1000 will be written to the data file and the following message will appear:

```
1000 has been entered
```

The second question is for specifying the total volume of the oil spill in U.S. gallons. This is the total volume, whether the spill is instantaneous or continuous. To answer this question, type in the value and then hit "Return."

The third question is for specifying the length of the time step for oil spill simulation. The default value is 15 minutes. If this value is desired, hit the "Return" key. If a different value is desired, type in the new value and then hit the "Return" key.

The next three questions are for specifying specific gravity of oil, kinematic viscosity of water (ft^2/s) and surface tension of oil (lb/ft), respectively. The default values for the type of oil that was defined previously are displayed. If the type of oil is not one of the three that were listed, an alternate default value will be shown. If the default value is desired, just hit "Return." If a different value is desired, type in the new value and then hit the "Return" key.

Once these questions are answered, a summary will be displayed on the screen to give you an opportunity to re-check the answers and correct any mistakes. The screen appears as follows (the numbers shown are for a sample case).

```
Total number of particles defined in the system: 1000

Total volume of oilspill (U.S. gal.): 5000

Length of time step for spill simulation: 15 min

Specific gravity of oil: 7

Kinematic viscosity of water: 1.411E-05 (sq ft/sec)

Surface tension of oil: .000755 (lbs/ft)

Is the above information correct? (Y/N)
```

If the information is correct, hit the "Y" key. If the information is not correct, hit the "N" key. If the latter is chosen, the series of questions will be repeated, giving you an opportunity to correct any mistakes. The next series of questions is as follows:

What is the X-coordinate of spill site (ft)?

What is the Y-coordinate of spill site (ft)?

What is the molar volume of oil? [.7063E-02] (cu ft/mol)

What is the solubility of fresh oil? Gasoline [0.873E-02] lbs/cu ft

What is the boiling point temperature of oil?
Note. Characteristic curves for the oils will be used if you enter a value less than 1.0 for the boiling point temperature. If you choose this option the value of coefficient C will be automatically be computed. Gasoline [311.0] deg.K.

What is the coefficient C of evaporation characteristics of oil?

Gasoline [6.11]

The first and second questions ask for the location of the oil spill. The coordinates of the spill can be found by going into the graphical routines and moving a marker around on the map where the spill took place. To answer these questions, type in the coordinates of the spill and then hit the "Return" key.

The third and fourth questions are for finding the appropriate values of the molar volume of oil (ft^3/mol) and solubility of fresh oil (lb/ft^3), respectively. The default values are shown in brackets after the question. If the default values are acceptable, just hit the "Return" key. If a different value is desired, type in the desired value and hit the "Return" key.

The fifth question asks for the boiling point temperature of oil in kelvins. Characteristic curves for crude oils will be used if you enter a value less than 1.0 for the boiling point temperature of oil. These characteristic curves are described in detail by Shen et al. (1990). If you enter a value less than 1.0, the sixth question in this series will not be asked but will automatically be computed. Each type of oil may have a different value. The default value for this question is shown between the brackets. If the default value is desired, hit the "Return" key. If a different value is desired, type in the value and hit the "Return" key.

The sixth question asks for the coefficient *C* of evaporation characteristics of oil. The default value for this question is shown between the brackets. If the default value is desired, hit the "Return" key. If a different value is desired, type in the value and hit the "Return" key.

After answering these questions, the following will appear on the screen:

X-coordinate of spill site is: 0 ft.

Y-coordinate of spill site is: 0 ft

Molar volume of oil is: .007063 cu ft/mol

Solubility of fresh oil is: .00873 lbs/cu ft

Coefficient C of evaporation characteristics of oil is: 6.11

Boiling point temperature of oil is: 311 deg K

Is the above information correct? (Y/N):

If the above information is correct, hit the "Y" key. If any of these values need to be changed, you prompt this set of questions again by hitting the "N" key.

After this series of questions has been completed, the following questions will be asked:

```
We are now entering data needed for the hour #1 wind and air temperature.

What is the wind speed (mph)? [ 0.0 ]

What is the wind direction (clockwise angle measured from North in degrees)?: [0.0]

What is the air temperature in deg. F.? [ 0.0 ]
```

These questions pertain to the first hour of wind data and air temperatures after the oil spill started.

The first question in this series is to determine the wind speed in mph. To answer this question, type in the desired value and then hit the "Return" key. If the default value is desired, just hit the "Return" key.

The second question asks for the wind direction. This is the clockwise angle measured in degrees from north. For example, the angle for wind out of the west is 270°. To answer this question, type in the desired value and then hit the "Return" key, or just hit the "Return" key to accept the default value.

The last question in this series is to determine the air temperature in °F. To answer this question, type in the desired value and then hit the "Return" key.

After these questions are answered, a summary of the data entered will be displayed on the screen for you to verify. The display on the screen will look as follows:

```
Data for hour #1 of simulation

Wind speed is: 5.0 mph (7.3 ft/s)

Wind direction is: 90 deg.

Air temperature is: 50 deg.F

Is the above information correct? (Y/N):
```

If the information is correct, hit the "Y" key. If you need to change any of the information, hit the "N" key. If the latter is chosen, the program will prompt you to answer this series of questions again. If the former is chosen, the program will ask the following question:

```
Would you like to "R"epeat the above values for hour #2 or enter "N"ew values? (R/N):
```

For every hour the oil spill simulation model will run, wind velocity and air temperature are needed. The program is designed in such a way that if the data for the second hour are the same as the previous hour, you can simply hit the "R" key If a different value is desired for one or more questions, hit the N key to enter new values for wind velocity and air temperature The program will repeat this question until all the data for the simulation period have been entered. When this is complete, the following messages will appear on the screen:

```
File STCL11.SPL has been saved.

Hit any key to continue
```

At this point all of the data for the data file with the extension of SPL have been created. These data are stored on the disk to be used later by ROSS or LROSS.

Subsection FLW

This subsection creates the data file with the extension FLW. This subsection contains the water level and discharge at each node in the river as defined by the one-dimensional flow model. Also included are the ice conditions at each cross section in the river. These data are independent from the ice region data in the ICE data file. This data file consists of three blocks of data. The first block contains a single entry, and it is for the time step in the one-dimensional model. The second block contains data for the discharge and water level. Ice conditions (ice thickness information) are in the third block of data. The program asks whether to create this data file or not by displaying the following on the screen.

```
This segment of the program creates the file with the extension FLW.
This file contains data needed to compute flow distribution in river.)

If you want to use one of the default flow data files (low, medium, high flow) you may skip this section.

Would you like to proceed in this section? (If you enter No, this data file will not be created.) (Y/N):
```

If you wish to create this data file, hit the "Y" key, or else hit "N" key. Upon entering this subsection, the following will appear on the screen:

```
What is the time step for river flow computation (hrs)
NOTE: This cannot exceed total simulation period.?
```

The question prompts you to enter the time step (hr) for the one-dimensional river flow computation. This value *must not* exceed the total simulation period as defined in subsection on SPL. To answer this question, type in the number of hours of this time step and hit the "Return" key. Suppose you typed "4" and hit the "Return" key. The computer will give you a chance to verify it by displaying the following message on the screen:

```
Is the time step for river flow computation 4.0 hrs.? (Y/N):
```

You can acknowledge the value by hitting the "Y" key. If the value is incorrect, hit the "N" key.

The next section pertains to the discharge and water level at the nodes of the river branches in the unsteady flow model. The questions are as follows:

```
Node No. 1

What is the water level? [45.34] ft

What is the discharge? [43432.3] cfs
```

The first question asks for the water level in feet above the reference datum at the node number as stated above. If the value in brackets after the question is correct, just hit the "Return" key. If this value is not correct, type in the correct value and then hit the "Return" key.

The second question asks for the discharge in cubic feet per second at the node as stated. If the value in brackets after the question is correct, just hit the "Return" key. If this value is not correct, type in the correct value and then hit the "Return" key.

After answering these questions, the program will check with you to see if the values that were entered are correct. The screen appears as follows:

```
Node No. 1

Water level is 45.34 ft

Discharge is 43432.3 cfs

Is this information correct? (Y/N):
```

If the data for each of the statements are correct, hit the "Y" key. If any of them are wrong, hit the "N" key to repeat the set of questions.

Once the data for a particular node are confirmed, the program will prompt you to enter the data for the next node. These questions will end once the water levels and discharges have been entered for all the nodes in the river. IPROCD knows the exact number of nodes in the river if you used one of the first four or five character sets recommended for the file name.

The next question will ask how many cross sections are ice covered. The question is:

```
What is the number of cross sections with ice covered conditions?
```

To answer this question, type in the number of cross sections that are ice covered and hit the "Return" key. If you are creating data for open water conditions, you may enter "0." The program verifies the value you entered, e.g. "3," by asking the following question:

```
Is number of cross sections with ice covered conditions: 3? (Y/N)
```

If this is the correct value, hit the "Y" key. You may correct the value by hitting the "N" key.

It is now necessary to identify each cross section. This is done by asking the following question:

```
What is the cross section no.?
```

You can answer this question by typing the cross section number and then hitting the "Return" key. To determine the ice cover condition, IPROCD will prompt you with the following question:

```
The cross section ice cover condition is: (1-FULL/2-PART/3-OPEN)
```

If the cross section is completely ice covered, select option 1. If the cross section is partially covered with ice, select option 2. If the cross section does not have an ice cover, select option 3. To answer this question, hit the number of the option that is desired. IPROCD will verify your entries by asking the following question (the illustration below assumes you that answered with "1"):

```
Cross section No. 1 has a(n) FULL ice condition.
Is this correct? (Y/N):
```

If the information that was entered is correct, hit the "Y" key. If you answered by hitting the "N" key, IPROCD will give you another chance to correct the previous entries.

At this point IPROCD will do one of the following, depending on the ice cover option that was selected.

Option 1 - Full ice cover. When this option is selected, you need to give only one ice thickness for the entire cross section. This a convenient feature when the ice cover across the cross section is uniform. If you have a full ice cover but it is non-uniform, you must select the "2-PART" option. When IPROCD enters option 1, the following will be displayed on the screen:

```
What is the thickness of the ice? (ft):
```

Answer this question by typing the value of ice thickness, e.g. "1.87," and then hitting the "Return" key. IPROCD will verify the value by displaying

```
Is the ice thickness 1.87 ft.? (Y/N):
```

If the value is incorrect, hitting the "N" key will give you another chance to specify ice thickness. If the value entered is correct, hit the "Y" key.

At this point the program can do two things. If there is another ice-covered condition, those data must be entered; the program will go back to the cross section number. This will depend on the value you specified for the number of cross sections with ice-covered conditions. You do not need to remember anything. IPROCD will prompt you until you have completed the data for all ice-covered sections. If the data for all the cross sections have been completed, IPROCD will ask if there is another time step. The display will be as follows:

```
Do you need time step 2 for river flow computation? (Y/N):
```

To enter into this loop, hit the "Y" key. You may enter the data for another time step similar to the one before. If another time step is not needed, hit the "N" key and the following will appear on the screen:

```
File STCL11.FLW has been saved.
Hit any key to continue.
```

At this point all of the data that are needed for this data file have been entered. Just hit any key to continue.

Option 2: Partial ice cover. The geometry of a river cross section is considered to be an assembly of trapezoids. Each cross section has a number of sounding depths (vertical lines) that define the cross section. To find the number of vertical lines that define the cross section, you must know the name of the river and the cross section number. The number of vertical lines and their distances from the left bank of the river can be found in Table A1 of Shen et al. (in prep., a).

```
How many sounding depths (vertical lines) define this cross section?
(See supplied table): ?
```

Suppose you answered by typing "5" and hitting "Return." IPROCD will verify your entry by displaying

```
Are there 5 sounding depths in this cross section? (Y/N):
```

You will get a second chance to correct the above value by answering "N" to the above question. Otherwise, hit the "Y" key.

The cross-sectional profile of the ice cover is defined by giving the ice thickness at the vertical lines. You have already entered the number of vertical lines that define the cross section. IPROCD will prompt you to enter ice thickness data for each of these vertical lines by displaying

```
What is the ice thickness (ft) at vertical line # 1?
```

Answer this question by typing the desired value (e.g. "1.05") and hitting the "Return" key. Then IPROCD will give you a chance to verify your entry by displaying

```
Is the ice thickness at vertical line 1: 1.05 ft (Y/N):
```

You may acknowledge the value by typing "Y." Typing "N" gives you a chance for error recovery. This sequence will be continued until data for all the vertical lines have been entered.

Option 3: Open ice condition. When this option is selected, the screen display will be

```
Cross section no. xx has a(n) OPEN ice condition.
Is this correct? (Y/N):
```

You can verify it by typing "Y." Typing "N" gives you the opportunity to redefine the ice conditions for this cross section.

Subsection ICE

This subsection creates the data file with the extension ICE. This data file consists of one data block, which contains information identifying the areal coverage of ice regions. The data can be changed for every time step of the one-dimensional flow model. If ice region data do not change from one time step to another in the one-dimensional flow model, the same data as the previous step must be entered for the present time step. An ice region is a range of grid boxes covered by ice. This information is used to determine whether spreading and advection take place under an ice cover or on open water. The program asks whether to create this data file or not by displaying the following on the screen:

```
This segment of the program creates the data file with the extension ICE.
(This file contains the areal coverage of ice.)

Would you like to proceed in this section? (If you enter No, this data file will not be created.) (Y/N).
```

If you wish to create or change this data file, hit "Y;" otherwise hit "N."

The first question in this subsection asks if you are running a simulation for an open water case. The question is as follows:

```
Are you running for an open water case? (Y/N)
```

If the oil spill simulation model will be run for an open water case, hit the "Y" key. If this is done, then the program will ask no more questions in this subsection. The program will display the following on the screen:

```
To continue hit any key.
```

If the oil spill simulation model will not be run for an open water case, then hit the "N" key.

The following parameters are needed for simulating oil spreading under ice:

```
What is Manning's n for ice roughness? [0.035]
What is the viscosity of oil? [0.84] lbs/ft sec
```

The first question asks for Manning's *n* for the ice cover. The default value for this question is inside the brackets after the question. The default value may be entered by just hitting the "Return" key. If a different value is desired, type in the value and hit the "Return" key.

The second question asks for the viscosity of oil. The default value for this question is inside the brackets after the question. Again you may follow the same procedure as for the first question.

Verification of these two entries is done by displaying

```
Manning's n for ice roughness: .035
Viscosity of oil: .84 (lbs/ft sec)
Is the above information correct? (Y/N):
```

If the information is not correct, hit the "N" key. This will give you a chance to re-enter the values. You may acknowledge the values by hitting the "Y" key.

The program branches off in two directions at this point. Very early in IPROCD a question was asked to determine if these data were to be created for ROSS or LROSS. The answer to this question decides which option the program will take.

ROSS Option

The following statement and question will appear on the screen at this point:

```
For the unsteady flow model time step: 1
What is the total number of ice regions?
```

These are the ice regions as seen from a plan view. A maximum of 20 is allowed in the current versions of ROSS and LROSS. If you answered this question by typing in a numb· e.g. "3") and hitting the "Return" key, the following will appear on the screen:

```
The total number of ice regions is: 3
Is this correct? (Y/N):
```

This is the verifying stage. Once again you can type either "Y" or "N." If you answered "Y," the following four questions will appear on the screen sequentially as you answ·· ·· ·i question:

```
Ice region no. 1
The x grid at the beginning of ice region?
The y grid at the beginning of ice region?
The x grid at the end of ice region?
The y grid at the end of ice region?
```

These four parameters define an ice region. Consult Shen et al. (in prep., a) for details. To answer these questions, type in the requested number and hit the "Return" key. After all four questions have been answered, the following summary will be printed on the screen for verification (the numbers for *x,y* on the illustration below are for a sample case):

```
Ice region no. 3
The x,y grid 15, 18 to the x,y grid 70, 28
Is the above information correct? (Y/N)
```

If the entered information is incorrect, hit the "N" key and the questions will be repeated. If the information is correct, hit the "Y" key. This set of questions will be repeated until data entering is complete for all ice regions. Once this is complete, the procedure can be repeated for another time step. At this stage the display on the screen will be

```
Would you like to enter data for another time interval? (Y/N):
```

If you answer by typing "Y," you will be asked to input ice region information for another time step. It is possible that ice regions exist only for some time steps. If this is the case, you must answer "Y" to the following question *x* number of times, where *x* is the number of time steps. (That means you must have ice region data that correspond to each time step of the one-dimensional model, regardless of whether the river has ice or not. However, if there is no ice, your entries are simpler.)

```
Would you like to enter data for another time interval? (Y/N)
```

Then for the time steps without ice, answer "0" to the question:

```
The total number of ice regions:
```

If you have given ice region information for all the steps, hit the "N" key and the following will appear on the screen.

```
File STCL11.ICE has been saved.
Hit any key to continue.
```

All the data for this file have now been entered and the file has been saved. Hit any key to continue with the program.

LROSS Option

In LROSS you need to specify lake ice regions as well as river ice regions. However, if you do not plan to run the simulation until the spill enters the river, you need not worry about the river ice regions. In the current version the total number of ice regions cannot exceed 20. The following statement and questions will appear on the screen at this point:

```
For the unsteady flow model time step: 1
What is the total number of ice regions?
What is the number of ice regions in lake?
```

Answer these questions by typing in the appropriate values and then hitting the "Return" key.

IPROCD will verify your entries by displaying (the numbers are for a sample case):

```
The total number of ice regions is: 3
The total number of ice regions in lake is: 2
Is this correct? (Y/N):
```

If the information entered is not correct, hit the "N" key to reprompt the questions. Otherwise, hit "Y" to continue.

At this point the following five questions will appear on the screen sequentially as you answer each question:

```
NOTE: Lake ice regions must be input first.

Ice region No. 1
The x grid at the beginning of ice region?
The y grid at the beginning of ice region?
The x grid at the end of ice region?
The y grid at the end of ice region?
The ice thickness in lake ice region (ft.)?
```

The first four parameters define the location of an ice region. If more details are desired, consult Shen et al. (1989b).

The fifth question defines the ice thickness for the region. This question will appear only when you are entering data for lake ice regions. To answer these questions, type in the requested number and hit the "Return" key.

When all five or four questions have been answered, a summary as shown in the following will be printed on the screen for verification:

```
Ice region No. 1
The x,y grid 28, 23 to the x,y grid 28, 33
The ice thickness in lake ice region is: 0.75
Is the above information correct? (Y/N):
```

If the information is incorrect, hit the "N" key and the questions will be repeated. If the information is correct, hit the "Y" key. This set of questions will be repeated until the data-entering process is completed for all ice regions. When you are entering data for lake ice regions, you will see five questions, after that only the first four questions will be asked for river ice regions. Once this is completed, the procedure can be repeated for another time step. At this stage the display on the screen will be:

```
Would you like to enter data for another time interval? (Y/N):
```

If you answer by typing "Y," ice region information can be repeated for another time step. It is possible that ice regions exist only for some time steps. If this is the case, you must answer "Y" to the following question x number of times, where x is the number of time steps:

```
Would you like to enter data for another time interval? (Y/N)
```

If you do not need to enter ice region data for another time interval, hit the "N" key and the following will appear on the screen:

```
File test.ICE has been saved.
Hit any key to continue.
```

All the data for this file have now been entered and the file has been saved. To continue with the program, hit any key.

Subsection BND

This subsection creates the data file with the extension BND. This data file consists of one data block, which contains half-life data for shorelines. The program asks whether to create this data file or not by displaying the following on the screen:

```
This segment of the program creates the file with the extension BND.
(This file contains shoreline half-life data.)

Would you like to proceed in this section? (If you enter No, this data file will not be created.) (Y/N).
```

If you wish to create or change this data file, hit "Y," otherwise hit "N." Upon entering this subsection the screen will display

```
Half life designation for shores: range no. 1
Shore number
1 = lower river
2 = upper river
3 = lower island
4 = upper island
Enter shore number code (1/2/3/4): [ 0 ]
```

First you have to select the shore number. There is a default value in brackets that follow the question. Just hit the "Return" key to accept the default value. If a different value is desired, type in the new value and then hit the "Return" key. When you have answered this question, IPROCD will ask for the beginning and ending boxes for this range by displaying sequentially

```
What is the beginning box no. for this range?
What is the ending box no. for this range?
What is the half life code to be assigned to this range (1-10)?
```

To answer these questions, type in the number and hit the "Return" key.

Once these questions are answered, a summary as shown in the following will appear on the screen for verification.

```
Half life designation for shores: range No. 1
Shore number 2 upper river
Half life designation to shore limits in x direction (Grid Box numbers)
Begins at 21, ends at 34
The half-life code assigned to this range: 4
Is the above information correct? (Y/N)
```

If the information that was entered is not correct, hit the "N" key to prompt the series of questions again. If the information is correct, hit the "Y" key. The program then asks if you would like to create data for another shoreline region. The question is as follows:

```
Do you wish to create another range of grid boxes? (Y/N):
```

If you hit the "Y" key, the previous set of questions will be repeated and you can create shoreline data for another range of grid boxes. If this is not desired, hit the "N" key.

At this point the data file for this subsection has been created and saved. The following will appear on the screen.

```
File STCL11.BND has been saved.
Hit any key to continue.
```

Subsection LAKEWIND.DAT

This subsection is needed only when you intend to run LROSS. If you are creating data for ROSS, the program will skip this section.

This subsection contains meteorological data used by the lake circulation model and has one block of data. There may be a maximum of 25 wind stations per time interval. If this subsection is entered, the following will appear on the screen:

```
This segment of the program generates the data file LAKEWIND.DAT
(This will be used in the lake circulation model.)

Would you like to proceed in this section? (If you enter No, this data file will not be created.) (Y/N):
```

If you wish to create or change this data file, hit "Y;" otherwise hit "N."

IPROCD enters this subsection only if you have indicated at the beginning that you wish to create data for LROSS. Upon entering this subsection the following eight questions will be asked sequentially:

```
The time at which the wind observation is made is (hrs.)?
The latitude of wind observation point is (deg. north)?
The longitude of wind observation point (deg. west)?
The height of instruments is (ft.)?
The temperature of air is (deg. F)?
The temperature of water is (deg. f)?
The wind speed is (mph)?
The wind direction is (deg. clockwise)?
```

The answer to the first question is the time elapsed after the oil spill began. Type in the value and then hit the "Return" key.

The second and third questions are to determine the latitude and longitude of the wind observation point. To answer these questions, type in the appropriate values followed by the "Return" key.

The fourth question is to find the height of the wind observation instruments above the water surface. Questions five to seven are self explanatory. The last question asks for the wind direction. This is the clockwise angle from north in degrees (e.g. wind out of the west = 270°). Answer these questions by typing in the appropriate value and hitting the "Return" key.

Once these questions are answered, a summary will be displayed as follows (the numbers shown below are for a sample case):

```
The time of observation is: hr 3.00
The latitude of wind observation point is: 44.00 deg North
The longitude of wind observation point is: 71 deg East
The height of instruments is: 10.00 ft
The temperature of air is: 40.00 deg. F
The temperature of water is: 60.00 deg. F
The wind speed is: 5.00 mph (7.33 ft/sec)
The wind direction is: 90.00 deg
Is the above information correct? (Y/N):
```

If any of the data in the above display need to be changed, hit the "N" key. If the display is correct, hit the "Y" key to proceed. At this point you will have the option to enter wind data for another station for the same time or the same station at a different time, or to terminate entering wind data. The following question will appear on your screen.

```
Would you like to enter data for another wind station or another time step?
(Y/N):
```

If you hit the "Y" key you can create a similar set of data that corresponds either to a different station or a different time. If you would like to terminate data entry, hit the "N" key.

All of the data for this file have now been entered and saved. The following will appear on the screen.

```
File LAKEWIND.DAT has been saved.
Hit any key to continue.
```

At this point, hit any key to continue with the program.

Defining file names and unit numbers

At this point in the program all but one data file have been created. This last data file* contains all the data file names and unit numbers that will be used in the oil spill simulation models. If you have created data files in this section, their file names will be used automatically by this program. These file names cannot be changed for this last data file. Names of all other data files which were not created during this session may be specified at this time.

The first file name is for the geometric data file. The geometric data for each of the rivers and lakes have already been set up. You only need to choose one from the list provided. For this file assignment the following will appear on the screen:

---

*For advanced users: The name of this file is either ROSS.FNM or LROSS.FNM depending on whether you are creating data for ROSS or LROSS.

Individual datafiles created (if any) during the session have been saved.
Now we are going to set up the file specification for corresponding unit numbers. [Unit numbers will be assigned by the program. You need to specify only the file names.]

For obtaining the geometric data file, please hit the number of the corresponding river/lake:

1 - St. Clair River
2 - Detroit River
3 - Upper St. Mary's River
4 - Lower St. Mary's River
5 - Lake St. Clair and Detroit River

To answer this question, just hit the number corresponding to the river or lake that you wish to simulate. The program will automatically assign the appropriate geometric data file name.

After answering this question, questions will appear on the screen for data files with extensions SPL, BND, FLW and ICE. If a data file was created during the execution of this program, no question for that file will appear.

The first question asks for the file name of the data that has the extension of SPL. The following will appear on the screen for this question:

For the file with the extension SPL, enter the 6 letter name:
(This file contains oilspill data) [STCL11]

To answer this question, either hit the "Return" key to accept the file name in the brackets or type in a different file name and then hit the "Return" key.

The second question asks for the file name of the data that has the extension of BND. The following will appear on the screen for this question:

For the file with extension BND, enter the 6 letter name:
(This file contains shoreline data) [STCL11]

To answer this question, either hit the "Return" key to accept the file name in the brackets or type in a different file name and hit the "Return" key.

The third question asks for the file name of the data that has the extension of FLW. The following will appear on the screen for this question. (Note that a set of 15 flow data files have been created. You may choose any one of these or any of your own flow data.)

For the file with the extension FLW, enter the 6 letter name.
The default file names for low, medium and high flow are as follows:

	HIGH FLOW	MEDIUM FLOW	LOW FLOW
St. Clair River	STCLHI Q = 230,000	STCLME Q = 188,000	STCLLO Q = 130,000
Detroit River	DETRHI Q = 210,000	DETRME Q = 184,000	DETRLO Q = 170,000
Upper St. Marys River	STMUHI Q = 110,000	STMUME Q = 76,000	STMULO Q = 57,000
Lower St. Marys River	STMLHI Q = 110,000	STMLME Q = 76,000	STMLLO Q = 57,000
Lake St. Clair & Detroit River	LDETRHI Q = 210,000	LDTERME Q = 184,000	LDTERLO Q = 170,000

If you choose from this table, enter the corresponding 6 letter name in the table above the flow discharge number [STCL11]

This table shows the river or lake and three possible flow conditions. There are default files that contain the data for these flow conditions. The default file names are above the discharge volume for the flow condition of the river or lake.

For example, the default file name for the lower St. Marys River with a flow condition of about 76,000 cfs is found as follows:

1. Find the river or lake in the left column
2. Go to the right in that row and find the closest Q value.
3. The default filename is just above this value.

For this example the closest Q value is 76,000 cfs. This is the medium flow condition. The default file name for this value is STMLME.

For the default file name, type in the file name and then hit the "Return" key. Otherwise, type in a different desired file name and then hit the "Return" key.

The fourth question asks for the file name of the data that has the extension of ICE. The following will appear on the screen for this question:

```
For the file with the extension ICE, enter the 6 letter name:
(This file contains areal coverage of ice). [STCL11]
```

To answer this question, either hit the "Return" key for the file name in the brackets or type in a different file name and then hit the "Return" key.

After answering this series of questions, the following message will appear on the screen.

```
The data creation program is now complete. Hit any key to continue.
```

The creation of data files is now complete for ROSS. The program will return to the main menu if you hit any key.

If you choose to create data files for LROSS, one more data file name needs to be assigned. This data file is LAKEWIND.DAT. After this is done, the program will return to the main menu if you hit any key.

Output files of ROSS/LROSS

The output files always have the same name regardless of the run. These file names are:

```
OILP.   OUT
VELCAR..OUT
VELSTR.OUT
XXXXSP.OUT
```

OILPRT.OUT file contains the geometric and shoreline data. This file is not generated unless you have selected this option from the menu during the creation of the SPL file.

VELCAR.OUT and VELSTR.OUT are files that contain the information on velocity distribution. The ROSS and LROSS programs are capable of producing these files, but you cannot do this from interactive mode. Consult the ROSS and LROSS manuals for details.

XXXXSP.OUT is the data file that contains the information needed to produce graphical output. XXXX is the abbreviated name for the river. For example, the output file for the St. Clair River will always be STCLSP.OUT. The other four names are DETRSP.OUT, STMUSPOUT, STMLSP.OUT and LDETRSP.OUT. As you can see, the file name is based on the river. Therefore, when you make several runs, normally you will be left with only the data file that corresponds to the latest run, and the previous output file generated will be lost.

Suppose you need to save all the output data files that correspond to each run. You can do so by going to the system level and then using the copy command. The syntax for this command is

```
COPY filename1 filename2
```

You should not use the Rename command to change names in this case. You can also delete unwanted files by using the Delete command from the system level. The syntax for this command is

DEL filename

Use this command carefully.

MENU FOR THE GRAPHICS PROGRAMS

This chapter supplements the information available on the screen for running the graphics menu. First, the standard notation used within the graphics program needs to be explained. A value listed in angle brackets in a question is the default value. If you just hit the "Enter" key instead of typing in a value, then the default value will be used. For example, "Enter Y max 80120?" indicates that the default Y-max value is 80,120 ft. If you would like to use this value, hit "Enter." Otherwise, type in the desired value and then hit "Enter."

Graphics menu

Upon entering the graphics menu the screen display will show

```
1) St. Clair River
2) Detroit River
3) Lower St. Mary's River
4) Upper St. Mary's River
5) Lake St. Clair and Detroit River
6) Exit "Return" to Main Menu

        Select option number and then hit return:? 1
```

In response to this menu, enter the number that corresponds to the river or lake that you would like to see plotted and hit "Enter." The next question the computer will ask is

```
What oil spill file would you like to look at STCLSP.OUT?
```

You should enter the file name that corresponds to the case that you would like to plot. The default file name you see on the screen will be one of the following: STCLSP.OUT, DETRSP.OUT, STMUSP.OUT, STMLSP.OUT or LDETRSP.OUT, depending on your answer to the previous question. These names correspond to the latest output file created by ROSS or LROSS for the particular river. If the default name is acceptable, just hit "Enter." If you would like to view the result of an earlier run, then you should enter the name of the output file that corresponds to that run.

After the oil spill file is supplied, you will be in the main graphics program.

Initial options

There are several parameters that can be defined so that a plot can be adjusted to suit your requirements. There are many parameters you have to supply to control the form of the plot. These include the area of the river or lake that you would like to see plotted, the time levels at which the oil slick is to be plotted, the labels, monochrome or color output, plotter or screen output, single display or sequential display of oil slick, or multiple time levels on the display or plot.

There is a file on disk that contains the default values for X-min, Y-min, X-max and Y-max. These values are the X and Y coordinates of the lower left corner and the upper right corner of the screen, respectively. Later in the program you have the option of changing these values if you desire.

Plotting area

The program will first display the last saved values for X-min, Y-min, X-max and Y-max. The program will ask you if you would like to change these values (the scale) by displaying. For example,

```
The plotting area is currently defined by
        X-min: 1000
        X-max: 100000
        Y-min: 20000
        Y-max: 82810
```

You have the option to change the scale after you have seen the plot by using the arrow keys or the "C" key. Would you like to change the scale N ?

The default answer is "No." If you would like to change these values, answer "Y" to the question. You will then be asked to enter the values for each of the variables. If you do not want to change a particular value, just hit "Enter." If you answer "Y" to this question, you will be prompted to enter the four values X-min, Y-min, X-max and Y-max by displaying the following questions sequentially:

```
        Enter X-min  1000  ? 70000
        Enter X-max  100000  ? 94000
        Enter Y-min  –20000  ? 4000
Suggested values for Y-max are:
        18595 for no distortion on screen
        20560 for no distortion on plotter
        or 82810,  the data-file default
        Enter Y-max  82810  ?
```

After you enter the first three values, some information will be displayed on the screen to help you select a value. The distorted plots are computed based on the physical size of the screen and paper plot.

Time steps

The next question is to specify the number of time steps for which the oil slick is to be plotted. This question is displayed as follows:

```
Please enter the number of time-steps you would like to see—
  (Enter –1 to see all time-steps or 0 to see just the river boundary)
Number of steps? 3
```

Suppose you answered this question by typing 3 and then hitting "Return." Questions will prompt you to specify the time levels at which you would like to have the oil slick plotted. The screen display will be

```
Please enter step number:? 1
Please enter step number:? 2
Please enter step number:? 3
```

In this case, the user has specified the step numbers 1, 2 and 3. The time that correspond to these steps will depend on the time step of the oil spill model and the time step frequency at which the data were written to the data file XXXXSP.OUT. For example, if the time step in the oil spill model was 15 minutes, and data were written to XXXXSP.OUT data file every time step, steps 1, 2 and 3 represent 15 min., 30 min. and 45 min., respectively, after

the spill. On the other hand, for a 15-minute time step, if the data were written every four time steps in ROSS or LROSS, the step numbers 1, 2 and 3 in the graphics program represent 15 min., 1 hr. 15 min., and 2 hr. 15 min. In any case the real time will appear at the bottom of the plot, both on the screen and on the plotter. It is not necessary that the step numbers be consecutive. You may skip steps by specifying, for example, 4, 7 and 13. However, the steps must be in increasing order, i.e. you may not request step number 4 and then step number 1.

There are two special values that you may find useful. a value of 0 will show only the shoreline and labels (if selected in later questions), and a value of –1 will display all the time steps.

Plot type

The next question is whether you would like to see the plot in monochrome or color. The monochrome plot represents the oil slick as an assembly of particles, whereas the color plot can show the distribution of oil according to various concentrations. The screen display for this question would be

```
Which would you like to see: Monochrome or Color?
Enter M or C    M  ?
```

You can answer this question by typing "C" or "M."

Multiple plots

If you requested to see more than one time step, the program will ask if you would like to have multiple time steps plotted on the same plot by displaying

```
Would you like to see multiple plots on the same graph   N  ?
```

If you want more than one time step to appear on the same screen, answer "Y." If you would like to see each plot on a separate screen (or paper), type "N." The default is "N."

If you answer "Y" (indicating that there can be more than one time step on the same screen), there is another option you may select later to control how your plot looks on the screen. With this option you may clear all previous steps on the screen and replot the current step. Details for this option are given in the section on Further Options.

Geographic locations

The next question is whether you would like to see geographic locations and their labels marked on your plots. The screen display for the question is

```
Do you want geographic locations and labels to appear   N  ?
```

You may answer with "Y" or "N" followed by "Return." The default is "N," and you may accept it by just hitting Return. If you are plotting the entire river in one plot, choosing this option may make the plot too crowded with labels in some areas.

Mile markers

This question is to determine whether you would like to see the river mileage markers and their labels on your plot. The screen display for this question is:

```
Do you want mile markers and labels to appear   N  ?
```

Again, you may answer with "Y" or "N" followed by "Return." The default is "N," and you may accept it by just hitting "Return."

Plotter

The last question the program will ask at this stage is whether you would like to send the plot to the plotter. The default is "No." If you want a copy on the plotter, please make sure that the plotter is turned on now (for plotter information, see the section on plotter operation in this manual). The screen display for this question is.

Do you want a plotter output N ?

Again you may answer with "Y" or "N." The default value is "N" and you may accept it by just hitting "Return."

Plotting process: monochrome

If you have selected a monochrome plot, no more questions will be asked before the program starts plotting. It takes a few minutes for the computer to read and process all the data before it starts plotting. It is best not to do anything with the keyboard during this time period. If you have selected more than one step to be plotted, the computer will pause after it finishes plotting the first step. You can hit any key to continue. Any time you are in the pause mode, there are a few additional options that you can execute to control the way the plot will look. Refer to the section under Further Options for more details on these additional options. Figure 3 is a sample output from a monochrome plot.

Data processing and plotting: color

If you selected the color option, more questions will be asked to determine the color classification. There will be approximately a 2-minute delay between the last question you answered and the next screen display. It is best not to use the keyboard during this period. There are eight colors available for the classification for both the screen option

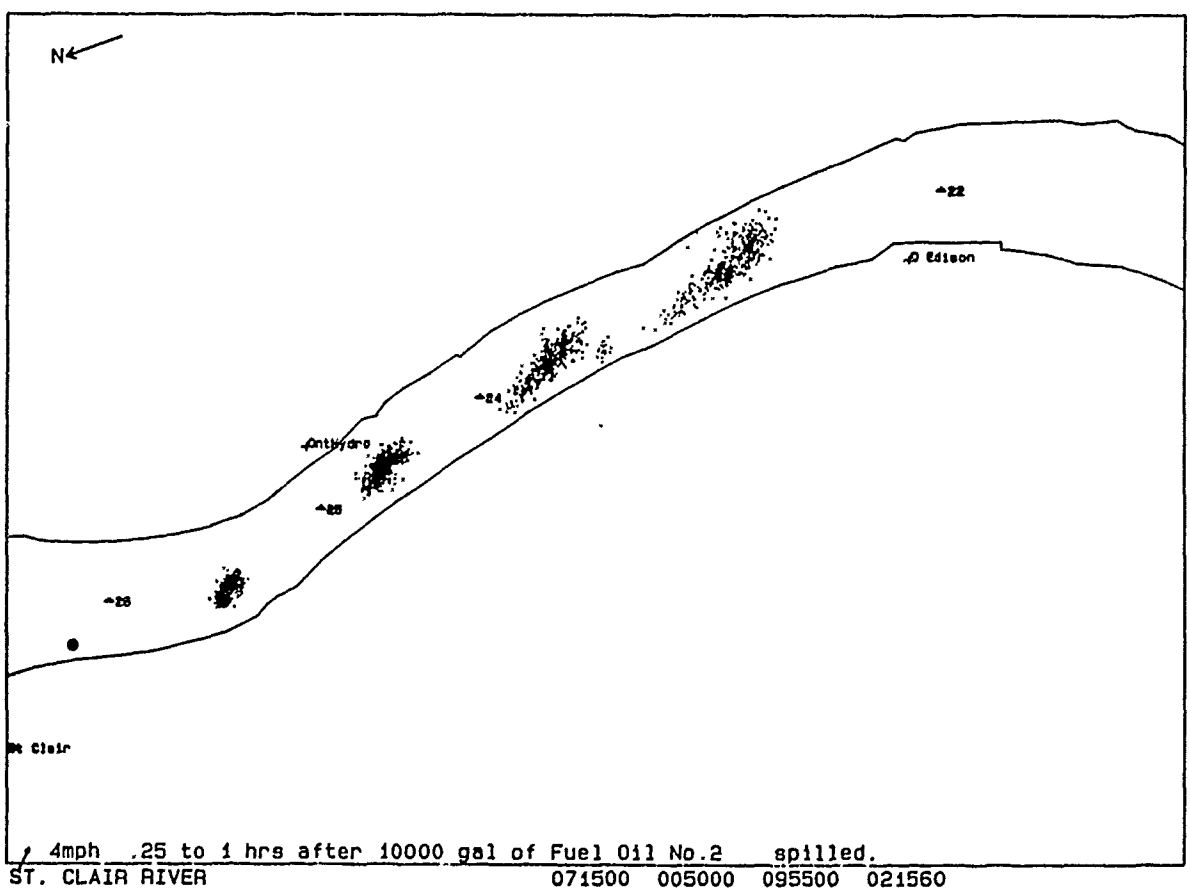


Figure 3. Monochrome plot for an instantaneous spill in the St. Clair River.

and the plotter option. Therefore, you need to specify the upper range for each color. Initially the program computes a linear classification based on the total volume available for the first time step to be plotted.

A sample display for a spill step containing 10,000 gallons is shown below.

```
Color Classification:
      500 Patches,   20 Gal/patch,  10000 Gal Total,   12 Boxes
        6 boxes with        1.0–        200.0 gallons
        0 boxes with      201.0–        400.0 gallons
        2 boxes with      401.0–        600.0 gallons
        0 boxes with      601.0–        800.0 gallons
        0 boxes with      801.0–       1000.0 gallons
        0 boxes with     1001.0–       1200.0 gallons
        1 boxes with     1201.0–       1400.0 gallons
        3 boxes with     1401.0–      10000.0 gallons

Would you like to change the Color Classifications  N  ?
```

The default answer is "N," which you can accept by just hitting "Return." If you typed "Y," you will be prompted to answer the following question sequentially:

```
Please enter the cut-off points for the color classifications.
Make sure that you type them in an ascending order:

Old value:  200.0  New value:?  50
Old value:  400.0  New value:? 100
Old value:  600.0  New value:? 150
Old value:  800.0  New value:? 200
Old value: 1000.0  New value:? 300
Old value: 1200.0  New value:? 350
```

The values shown after the question mark are for a sample case. After you enter the last value, the computer will take a few seconds to reprocess the color classification and then it will display the new classification as

```
Color Classification:

      500 Patches,  20 Gal/Patch,  10000 Gal Total,   12 Boxes
        3 boxes with        1.0–         50.0 gallons
        1 boxes with       51.0–        100.0 gallons
        0 boxes with      101.0–        150.0 gallons
        2 boxes with      151.0–        200.0 gallons
        0 boxes with      201.0–        300.0 gallons
        0 boxes with      301.0–        350.0 gallons
        0 boxes with      351.0–        400.0 gallons
        6 boxes with      401.0--     10000.0 gallons

Would you like to change the Color Classifications  N  ?
```

The procedure to answer this question is the same as before. You can continue with this process until you are satisfied with the color classification. If you answer "N" to this or just hit "Return," the computer will go into the plotting mode. After one step is plotted, the computer will pause until you hit any key. There are additional options that you can execute during this pause. Refer to the section on Further Options for details on these additional options.

If you selected multiple plots, the same color classification will be maintained from step to step to maintain the consistency. The total spill volume may not be the same from step to step due to evaporation and dissolution.

At the end of the plotting session, questions will be asked to determine if you need more plots of this spill. You may answer with "Y" or "N" to these questions. These questions are self-explanatory.

Further options

In addition to the options that were discussed, the program allows some further options. It is not necessary to know these options in order to run the program. However, they give more flexibility in controlling the output.

The program will plot one time step of the requested plot and display a message at the bottom of the screen indicating that the program is pausing and that you should hit a key to continue. This is considered the pause mode. There are several options available at this level to present the plot in different forms.

For this section of the program to operate correctly, you must make sure that the "numlock" light on the upper right side of the keyboard is on. If it is not on, turn it on.

If you would like to save the current X-min, Y-min, X-max and Y-max values, hit "S" when you are in the pause mode. This saves those values in the default value file. The advantage here is that if you plan to make many plots with the same window, you do not need to specify them each time once the values are saved.

To exit the program, hit "Q" for quit.

To clear the display and have the last time step replotted, hit "Home" (which is the "7" key on the numeric keypad). You can use this feature to erase unwanted plots on the screen.

If you would like to change the X-min, Y-min, X-max and Y-max values, hit the "C" key. The program will then prompt you for new values and will replot the last time step. This option can also be used to zoom in and zoom out. Generally, the next option is easier for zooming in. Therefore, "C" option is usually used for zooming out.

Zooming

If you would like to zoom in on the plot (to enlarge the spill area on the plot), just use the arrow keys located on the numeric keypad. When you hit an arrow key for the first time, a cross-hair cursor will appear at the center of the screen. This cross-hair has a light green color. You may move this cursor with the arrow keys. Move this cursor to the upper right corner of the area you would like to see enlarged. When you have the cursor at the position that you desire, hit "Enter." When you hit "Enter," you will see a violet cursor permanently marked at that position. You will also see another light green cross-hair cursor appear at the center of the screen. Move this cursor to the lower left corner of the area that you would like to see enlarged. These two cursors form an imaginary rectangle around the area to be zoomed in on. When you have the second cursor in the appropriate position, hit "Enter."

The program will clear the screen, zoom in on the area specified and replot the last time step. Note that when you zoomed in, you changed the X-min, Y-min, X-max and Y-max values.

Finding X,Y coordinates of a point

When you move the cross-hairs during the zooming option, you will notice that the numeric values that correspond to X-min, Y-min, X-max and Y-max at the bottom of the screen change. Therefore, this feature can be used to determine the coordinates of a spill location. In the first instance you will realize that your resolution is limited because the cursor moves a fixed distance for each hit of the cursor key. However, you can improve the resolution by zooming in and relocating the point until the required accuracy is achieved.

Brief explanation of the plots

Some of the descriptions shown on the plot (Fig. 3) will be explained here. On the top left corner you will see an arrow and an "N." This indicates magnetic north. On the bottom left corner you will see a small arrow. This shows the direction of the wind. These arrows show the correct direction only if the plotting scales are undistorted (i.e. Y-scale = X-scale). Next to the wind direction arrow the wind speed is printed in mph. The time that corresponds to the spill stage shown is also printed on that line. If you plot more than two time steps, only the times that correspond to the first and last step will appear on the plot. This was enforced to keep the descriptions on the plot to a desired level.

In addition, there are four numbers that appear below the margin. These numbers represent the coordinates of the limits of your current viewing area in feet. The sequence is X-min, Y-min, X-max, Y-max.

The spill site is indicated by a circle with cross hairs. In the color plot mode the physical size of the color blocks can be determined by the following formulas:

$$X = (X\text{-max} - X\text{-min}) * 0.011 \text{ (ft)}$$

and

$$Y = (Y\text{-max} - Y\text{-min}) * 0.0174 \text{ (ft)}.$$

A summary of commands to be used in the pause mode is given in Table 1.

Table 1. Summary of commands to be used in the pause mode. These are the commands that can be used when the program pauses with a plot on the screen. (Remember that the "numlock" light on the upper right side of the keyboard must be on.)

Arrows	Zooming features
C	Allows you to enter new values for X-min, Y-min, X-max, Y-max
Home	Clear the screen and replot last time step
Q	Quit
S	Save X-min, Y-min, X-max, Y-max in default value file

Using the plotter

To use the plotter, make sure that it is turned on. The power switch is on the back of the machine near the left rear corner. You will be able to hear the fan when the machine is on. The program is written and the computer is interfaced for an HP7550 plotter. Other HP7000 series plotters may also work successfully.

Paper

There are two ways to load paper into the plotter. The first and easiest way is to remove the paper tray in the front center portion of the machine by pulling it firmly. This paper tray is much like a photocopier paper tray. Fan the edge of the paper. The second way is to load paper manually. Place a sheet of paper under the rollers in the center of the plotting surface. Make sure that the paper is flush against the left side of the plotting surface. Press the "Load/Unload" button on the plotter until "*" appears in the upper right corner of the plotter display. Now you can automatically load paper just by hitting the "Load/Unload" button on the plotter.

The second way is to load paper manually. Place a sheet of paper under the rollers in the center of the plotting surface. Make sure that the paper is flush against the left side of the plotting surface. Press the "Load/Unload" button on the plotter. The plotter will grab the paper and move it into place.

Pens

To load pens into the plotter, flip up the hinged smoked plastic lid. On the right side of the plotting surface in a small well is the plotting carousel. Lift the carousel by gently pulling down on the plunger directly under the carousel position that you are loading. Slide the pen into the carousel and gently release the plunger. When you have loaded all eight carousel positions, put the carousel back into the small well you got it from. Lower the hinged cover. The plotter may ask you to hit the "Enter" button on the plotter before it will plot. If it does, just hit the "Enter" button on the plotter.

Problems

If you have any problems with the plotter, consult the two plotter manuals.

LITERATURE CITED

Shen, H.T., P.D. Yapa and M.E. Petroski (1990) Simulation of oil slick transport in Great Lakes connecting channels. Theory and model formulation. U.S.A. Cold Regions Research and Engineering Laboratory, CRREL Report 90-1.

Shen, H.T., P.D. Yapa and M.E. Petroski (In prep., a) Simulation of oil slick transport in Great Lakes connecting channels. User's manual for the River Oil Spill Simulation model. U.S.A. Cold Regions Research and Engineering Laboratory, Special Report.

Shen, H.T., P.D. Yapa and M.E. Petroski (In prep., b) Simulation of oil slick transport in Great Lakes connecting channels. User's manual for the Lake-River Oil Spill Simulation model. U.S.A. Cold Regions Research and Engineering Laboratory, Special Report.

U.S. GOVERNMENT PRINTING OFFICE 1991/500-063/42020

APPENDIX A. HARDWARE AND SOFTWARE REQUIREMENTS

For all elements of the program to work, it is necessary to have the following hardware:

- An IBM PC-AT-style computer (80286, 80386, 80486) with 640 K memory;
- A VGA graphics card;
- A VGA monitor;
- An HP 7550 plotter;
- A printer;
- A HALO* graphics subroutine package; and
- A math co-processor (80287 or 80387).

There are parts of the program that will not work if certain requirements are not available. These requirements are:

- If HALO is not available, the graphics part of the program will not work.
- If an HP plotter is not available, everything else will work except that hard copies of plots cannot be obtained.

---

* HALO is the trademark of the Media Cybernetics Co. Software rights must be obtained before installing this program on your machine.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	July 1991	

4. TITLE AND SUBTITLE
Simulation of Oil Slick Transport In Great Lakes Connecting Channels: User's Manual for the Microcomputer-Based Interactive Program

5. FUNDING NUMBERS
DACA33-85-C-0001

6. AUTHORS
Poojitha D. Yapa, Ronald J. Thomas, Jr., Randall S. Rutherford and Hung Tao Shen

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
U.S. Army Cold Regions Research and Engineering Laboratory
72 Lyme Road
Hanover, N.H. 03755-1290

8. PERFORMING ORGANIZATION REPORT NUMBER
Special Report 91-8

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Department of Transportation
Federal Aviation Administration
Research and Development Service
Washington, D.C. 20591

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

Available from NTIS, Springfield, Virginia 22161.

12b. DISTRIBUTION CODE

13. ABSTRACT (*Maximum 200 words*)

The growing concern over the impacts of oil spills on aquatic environments has led to the development of many computer models for simulating the transport and spreading of oil slicks in surface water. Almost all of these models were developed for coastal environments. In this study, two computer models, named ROSS and LROSS, were developed for simulating oil slick transport in rivers and lakes, respectively. This report explains how to use the microcomputer-based versions of these two models.

14. SUBJECT TERMS
Computer models
Great Lakes connecting channels
Oil slicks
Oil spills

15. NUMBER OF PAGES
37

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL